NASA/CR-97- 205756

FINAL
IN-20-CR
OCIT
063443

Final Report
NASA Grant No. NASA-NCC8-67
NASA Marshall Space Flight Center
Marshall Space Flight Center, AL

Past experience with hybrid rockets has shown that certain motor operating conditions are conducive to the formation of low frequency pressure oscillations, or flow instabilities, within the motor. Both past and present work in the hybrid propulsion community acknowledges deficiencies in the understanding of such behavior, though it seems probable that the answer lies in an interaction between the flow dynamics and the combustion heat release.

Knowledge of the fundamental flow dynamics is essential to the basic understanding of the overall stability problem. The present work represents a first step in this direction. A series of tests were conducted at NASA Marshall Space Flight Center (MSFC) on a laboratory-scale two dimensional water flow model of a hybrid rocket motor. Principal objectives were: (1) visualization of flow and measurement of flow velocity distributions: (2) assessment of the importance of shear layer instabilities in driving motor pressure oscillations; (3) determination of the interactions between flow induced shear layers with the mainstream flow, the secondary (wall) throughflow, and solid boundaries; (4) investigation of the interactions between wall flow oscillations and the mainstream flow pressure distribution.

The test bed was a ⅓ scale model of an 11 inch (28cm) subscale Solid Rocket Combustion Simulator (SRCS) hybrid motor which has recently undergone (hot-fire) tests at MSFC. The test section was installed in a dual closed-loop water flow facility, originally developed in 1990 to investigate fluid flow through porous materials. Oxidizer, or main stream flow, was simulated by a water stream entering through injector slots located in the forward section of the "motor"; this flow could be injected either axially (along the motor axis) or radially. Fuel grain "burning" (mass injection only) was simulated by a secondary water stream entering the test section through 100μ sintered bronze porous plate material. The test section side walls were constructed of clear acrylic for flow visualization and optical velocity measurements. Flow visualization was accomplished through injection of very small helium bubbles into the mainstream and/or secondary flow streams and recording their movement with a 1000 frames/second video recorder.

Results of these tests were presented at the 32nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference in Orlando, FL, in July 1996, in a paper entitled "Cold-Flow Study of Hybrid Rocket Motor Flow Dynamics". This paper (AIAA 96-2843) is included as Appendix 1 of this report.

APPENDIX 1: Cold-Flow Study of Hybrid Rocket Motor Flow Dynamics (AIAA 96-2843)

AIAA

AIAA 96-2843
Cold-Flow Study of Hybrid Rocket Motor Flow Dynamics

R. M. Jenkins
Auburn University, AL

W. D. Cruit and A. W. Smith
NASA Marshall Space Flight Center
Huntsville, AL

32nd AIAA/ASME/SAE/ASEE
Joint Propulsion Conference
July 1-3, 1996 / Lake Buena Vista, FL

Cold-Flow Study of Hybrid Rocket Motor Flow Dynamics

R. M. Jenkins*
Auburn University, AL

W. D. Cruit† and A. W. Smith‡
Marshall Space Flight Center
MSFC, AL

Abstract

A study is being conducted at NASA's Marshall Space Flight Center (MSFC) on a laboratory scale two-dimensional water flow model of a hybrid rocket motor. The test bed is a scaled model of a small hybrid motor which has recently undergone (hot-fire) tests at MSFC. The test section is installed in a dual closed-loop water flow facility, originally developed in 1990 to investigate fluid flow through porous materials. Oxidizer, or mainstream flow, is simulated by a water stream entering through injector slots located in the forward section of the "motor"; this flow may be injected either axially (along the motor axis) or radially. Fuel grain "burning" (mass injection only) is simulated by a secondary water stream entering the test section through 100μ sintered bronze porous plate material. The test section side walls are constructed of clear acrylic for flow visualization and optical velocity measurements. Principal objectives include: (1) visualization of flow and measurement of flow velocity distributions; (2) identification of large-scale shear layer structures within the flow; (3) determination of the interactions between flow induced shear layers with the main stream flow, the secondary (wall) through-flow, and solid boundaries; (4) investigation of the interactions between wall flow oscillations and the mainstream flow pressure distribution. The present paper describes this study and presents some preliminary results for (1)-(3).

*Associate Professor, Aerospace Engineering; Senior Member AIAA
†Aerospace Engineer, Combustion Physics Branch
‡Instrumentation Engineer, Fluid Dynamics Branch

Introduction

Recent hybrid motor testing at NASA's Marshall Space Flight Center (MSFC) has shown that certain motor operating conditions have produced pressure traces with significant oscillations, or flow instabilities. Analysis of data from such tests shows both high and low frequency pressure oscillations. The high frequency oscillations seem to indicate a mechanism driven by the first longitudinal acoustic mode of the combustion chamber[1,2,3]. Low pressure oscillations, on the other hand, seem to be driven by a different mechanism or set of mechanisms. For gaseous oxygen hybrid motors, mechanisms such as chuffing, some boundary layer phenomenon which couples the combustion response to externally imposed pressure oscillations, and vortex shedding from the fuel grain face have been suggested as causes of these low frequency pressure oscillations[1,4]. Other investigations have suggested that the flush/fill time may play a role in sustaining oscillations arising from another source[4,5] As a result of the testing done at MSFC, the question remains: what causes the initial oscillations? Chuffing has not proven to be a significant mechanism in the tests conducted at MSFC, primarily due to the nature of the oscillations and to post-firing grain inspection. [1] The role of fluid dynamics with regard to initiating pressure oscillations has been considered, specifically with respect to flow fields created behind rearward facing steps as in dumps (in the aft cavity) and flame holders (at the forward end of the grain). Periodic vortex shedding and downstream interactions have been given credit for driving pressure oscillations in solid propellant motors[6].

One approach to resolving the conflict over the driving force for these low frequency pressure oscillations is to better understand the subject: the hybrid motor. A first step in this direction is a series of studies being conducted at MSFC on a laboratory-scale two dimensional water flow model of a hybrid rocket motor. Such water flow studies have been previously performed elsewhere by Schadow, *et. al.*[7], with regard to similar phenomena associated with ramjet flow instabilities.

Test objectives include: (1) visualization of flow and measurement of flow velocity distributions; (2) identification of large-scale shear layer structures within the flow; (3) determination of the interactions between flow induced shear layers with the main stream flow, the secondary (wall) through-flow, and solid boundaries; and (4) investigation of the interactions between wall flow oscillations and the mainstream flow pressure distribution.

Test Facility and Model Description

The test facility was originally designed to study flow through porous material to simulate solid rocket motor fuel burning in the Advanced Solid Rocket Motor (ASRM) development program at NASA's MSFC. The test facility consisted of two 500 gallon (1892 liter) supply tanks with one 800 gallon (3028 liter) return tank and a two-dimensional test section. Both supply tanks were elevated to the same height and regulated through water supplied from the return tank through a variable speed pump to maintain equal elevation heads in each tank. One supply tank contained water which entered the test section vertically through a porous wall, while the other tank supplied water which entered the test section (unimpeded) horizontally. The two streams simulated wall burning and mainstream gas flow, respectively. The water flow was seeded with florescent dye for flow visualization. This concept has been adapted for simulation of the flow in a hybrid rocket motor.

The configuration selected for study is similar to motors used in a solid rocket motor simulator program during a series of tests performed by Thiokol to investigate suppression of pressure oscillations in subscale hybrid rocket motors.[1] This configuration consisted of three main circular port (CP) sections of fuel butted together with a forward section (upstream plenum) which was either lined or unlined with fuel and an aft section (downstream plenum) which was unlined. The ability to inject oxidizer radially into the aft end was incorporated. Post-firing weighing of each of the circular port sections revealed a variation in the oxidizer-to-fuel (O/F) ratio along the motor (as expected). This aspect of the hybrid was included in the model. Though several injectors were used, only radial injection patterns and axial injection patterns are incorporated in the model. Also included in Thiokol's study of pressure oscillation suppression, as well as in the model, were inhibitor plates or rearward facing steps of various heights which would inhibit burning at the face of the fuel grain and change the flow pattern at the head end of the motor.[1] This study included both "stable" and "unstable" motors; unstable motors had pressure oscillations higher than 5% of the average motor pressure.

The original constraints for the hybrid flow simulation model were set to accommodate minimal changes in the test facility while maintaining the ability to adequately model the entire geometry of a hybrid motor (albeit two-dimensionally). This led to a maximum test-bed length of approximately 36 inches (92 cm). In order to visualize and record phenomena in the entire test section, a minimum CP flowpath height (inner wall to inner wall) was set at 2 inches (5 cm). Based on data gathered from original testing in the ASRM model, 1/4 inch (0.64 cm) thick 100μ sintered bronze was selected to simulate mass addition through the walls (top and bottom). A preliminary model of the test facility and test rig was used to derive flow losses in the system and to determine achievable Reynolds number (Re) levels as compared to typical Reynolds numbers found in actual hybrid motors. The results are shown in Fig. 1, plotted as Re vs. scaling factor. As expected, strict Reynolds number similarity is not achievable with this model.

Constraints and requirements for the hybrid flow simulation model, as determined by the test objectives, were as follows:

1. *easily changeable configurations*
2. *flow visualization capability*
3. *laser doppler velocimetry (LDV) capability*
4. *particle displacement tracking (PDT) capability*
5. *O/F ratio similar to actual motor*
6. *geometry simulation of actual motor including forward and aft steps*
7. *various sections of "fuel burning"/mass addition (3 piece circular port and head-end lining)*
8. *various areas of "oxidizer" injection (injector and aft end)*
9. *oscillating secondary flow (to assure an "unstable" configuration)*
10. *inlet and exit areas simulating actual motor (injector and nozzle geometry)*

NASA/MSFC Dual-Loop Water Test Rig

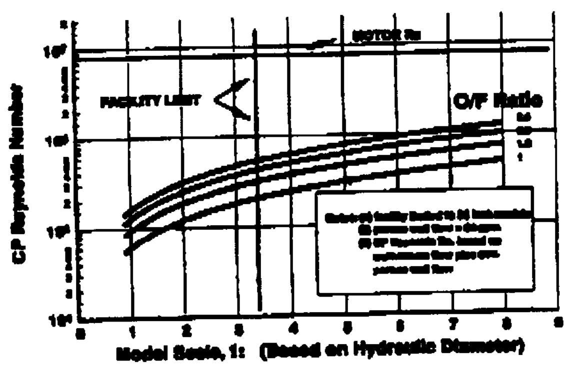


Fig. 1: Facility Reynolds number study

The model scaling rationale was as follows:

- Initial scale was based on the actual motor size and the test facility limits for model length.
- Model size chosen was a trade-off between Reynolds number scaling (small scale desired) and ease of visualization and measurement (large scale desired).
- The scale factor and the CP diameter of

the actual motor fixed the model "CP" hydraulic diameter.

• A 2.0 inch x 0.875 inch (5 cm x 2.2 cm) rectangular CP passage was selected to satisfy the hydraulic diameter criterion. The 5 cm passage height was chosen for ease of flow measurement.

• The upstream plenum configuration resulted from the constraints of hydraulic diameter (2.33 inches or 5.9 cm), step height scaling, and scaling of Reynolds number change between the plenum and the CP portion. The result was a 3.12 inch x 1.86 inch (7.9 cm x 4.72 cm) rectangular passage.

• The downstream plenum has the same (throughflow) dimensions as the upstream plenum.

The lengths for the upstream plenum, CP portion, downstream plenum, and nozzle ramp out sections were scaled directly from the actual motor dimensions. The model nozzle angle was the same as in the actual motor. The injector opening area was not scaled from actual motor injector data due to limitations in the available test facility pressure head. Since the injector produces the largest flow restriction, injection area was determined from O/F requirements and pressure drop. The ratio of the required open area of the inhibitor plates to the CP flow area was determined based on the size of the inhibitor plates used in the Thiokol tests. Three inhibitor plates were made, with open area ratios of 20%, 36% and 64% (as was a fourth plate, with 100% open area). To provide a means to externally oscillate the secondary or wall flow, a modified systolic pump was installed in the feed line for that loop. The pump was connected to a variable speed drive which allowed for forced frequencies from 6 Hz to in excess of 25 Hz. None of the results reported here utilized the external oscillation system, however. Schematics of the model and test facility are shown in Figs. 2 and 3, respectively.

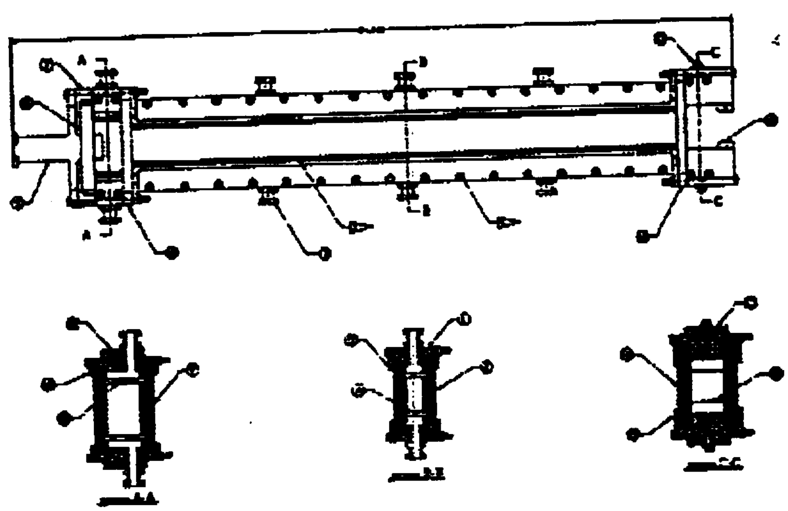


Fig. 2: Model Schematic

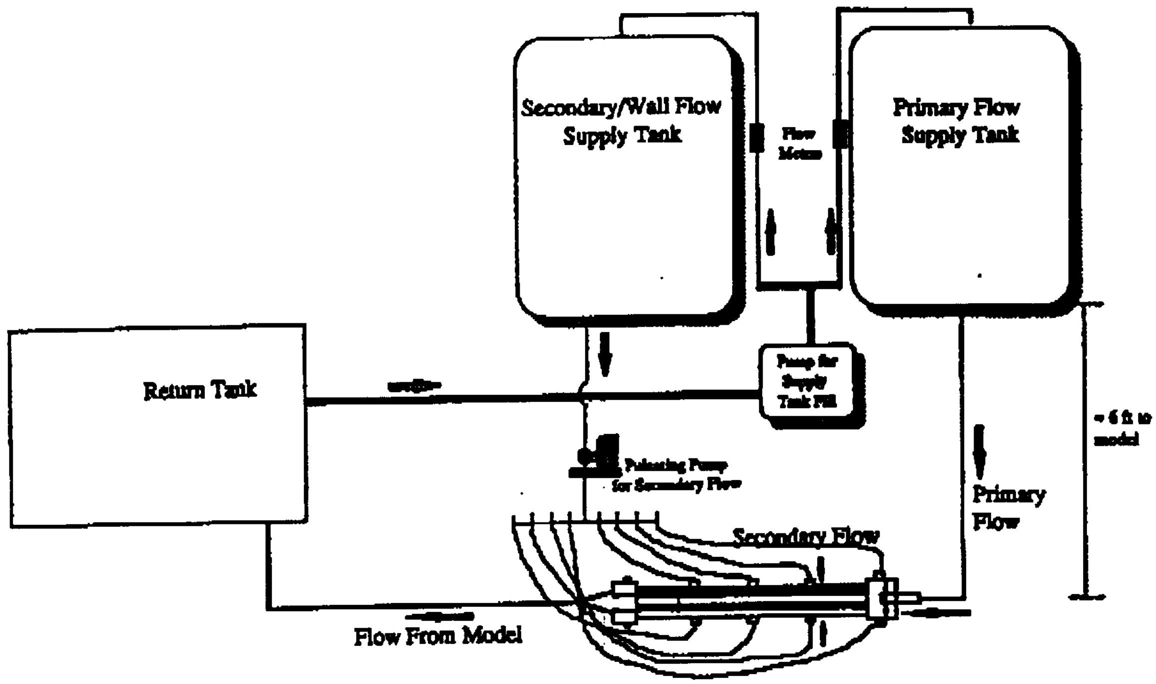


Fig. 3: Test Facility Schematic

Instrumentation and Test Procedure

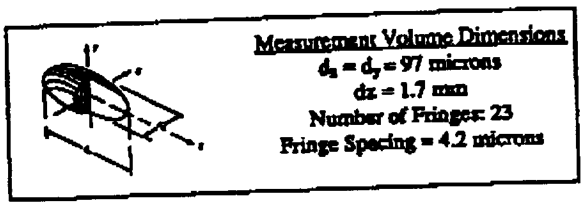
Four test configurations were studied in this phase of the project. *Configuration 1*, the baseline configuration, consisted of the model with the primary mass flow ("oxidizer") injected axially into the upstream plenum and secondary mass flow ("fuel") injected through the upstream plenum walls and through all sections of the CP portion. The 100% open plate was used (no inhibitor), and no aft end (downstream plenum) secondary wall flow was utilized. Preliminary testing for this configuration included seeding the flow with laser sensitive dye and recording the head and aft sections at both 500 and 1000 frames per second (fps). The upstream plenum secondary flow was then seeded with helium bubbles and back-lighting was used to do flow visualization at 1000 fps. Following this, LDV measurements were made.

Configuration 2 utilized the baseline configuration except the radial injector was incorporated. Testing included flow visualization with laser sensitive dye, flow visualization with helium bubbles, and LDV measurements.

In *configuration 3* the 64% open area flow inhibitor plate was installed in the baseline configuration (axial injector). Testing with this configuration included flow visualization with helium bubbles and LDV measurements. It was determined after processing the flow visualization data that the laser sensitive dye added no real value to the study being conducted.

Configuration 4 was the same as configuration 2 (radial injector) except that the 64% open area flow inhibitor plate was installed. Flow visualization with helium bubbles and LDV measurements were made.

LDV measurements were made with a TSI Inc. fiber optic based system. Shown schematically in Fig. 4, the system uses a 300 milli-Watt (mW) Argon Ion Laser that results in approximately 50 mW of laser power in the measuring volume. The optical signal is amplified and processed by TSI's ColorLink and IFA 750 respectively. A fifteen millimeter diameter probe was used, providing excellent measurement volume spatial resolution (see Fig. 4). Aluminum oxide particles of 1 micron diameter were added to the water supply tank for LDV seeding. These particles are excellent for water because they mix easily and are inexpensive. (Union Carbide Corporation Specialty Powders Business, 1555 Main Street, Indianapolis, IN 46224).



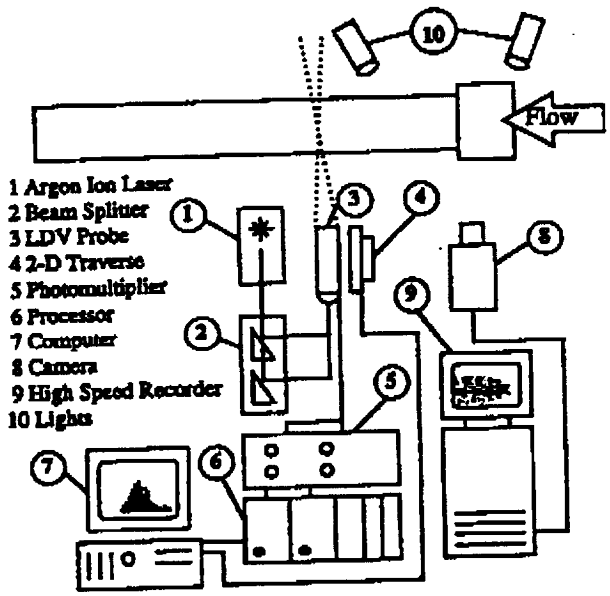


Fig. 4: LDV system schematic

Velocity measurements were made in both the upstream plenum and CP portions of the model. The measuring volume was centered in the model by visual inspection, and probe movements were accomplished with computer control of a traverse table. Excellent data rates were maintained by stirring up the particles that settled in the storage tanks. A high speed video system, NAC's HSV-1000, was used to generally characterize the upstream plenum and CP flow fields. Helium bubbles were injected into the oxidizer flowstream and illuminated from the rear. In addition to the LDV measurements, video pictures were made at 500-1000 fps.

Results

Slow motion replay of the high-speed video clearly shows generation of vortices (in most instances) from the inhibitor simulator and flow recirculation zones in the head end for both radial and axial injectors. Shedding frequencies, as obtained by visual count of vortices over a specified time interval (~ 0.4 seconds, real time), are as follows:

Configuration 1: not discernable
Configuration 2: 35 Hz
Configuration 3: 30 Hz
Configuration 4: 39 Hz

Estimated uncertainty in the above values is ± 20%. Results for configuration 1 were too uncertain to report. These frequencies are in the expected range for shed vortex pairs with a Strohal number of the order 0.25 - 0.50.

Figures 5 - 12 show the LDV results for each configuration. In each case, approximately 1200 data points were taken (both u and v velocity components). Data were then interpolated and plotted in either streamline format or as absolute velocity contours (the square root of the sum of the squares of the velocity components).

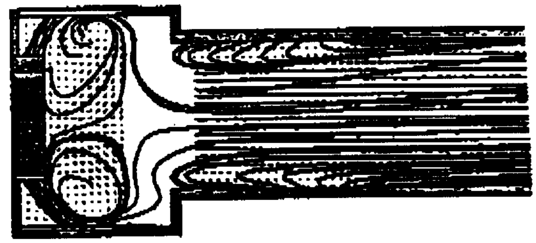


Fig. 7: Configuration 2 LDV streamlines

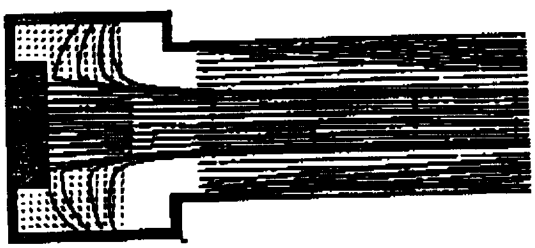


Fig. 5: Configuration 1 LDV streamlines

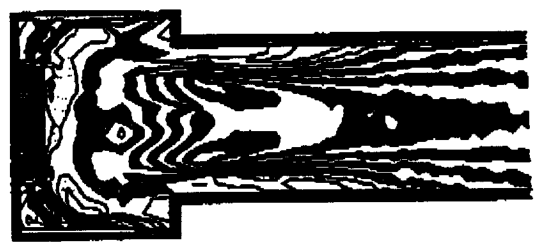


Fig. 8: Configuration 2 LDV velocity contours

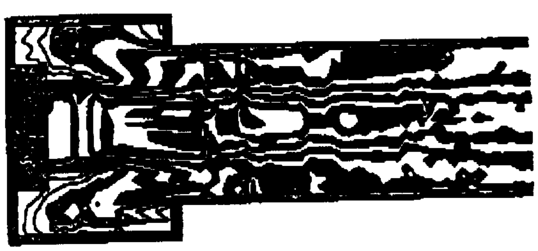


Fig. 6: Configuration 1 LDV velocity contours

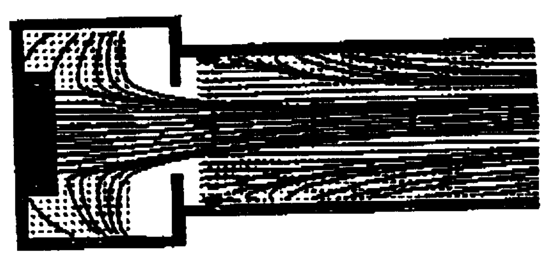


Fig. 9: Configuration 3 LDV streamlines

Clearly seen in the data are such features as the injected mainstream flow, the injected wall flow, and, in some instances, large-scale recirculation regions.

Finally, Figures 13 and 14 show the influence of the 64% open inhibitor plate on the overall (time averaged) flow. This is done by taking the difference between the data sets obtained for the 100% open and 64% open cases and plotting the results as absolute velocity contours. Local velocity differences range up to about 9 ft/sec (2.75 m/sec). For clarity, differences less than 1 ft/sec (0.3 m/sec) are not shown.

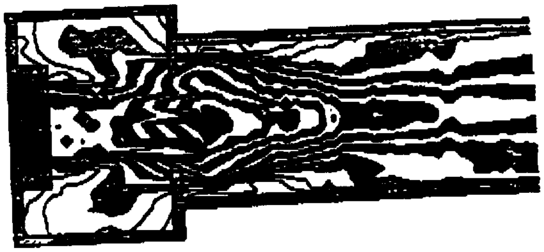


Fig. 10: Configuration 3 LDV velocity contours

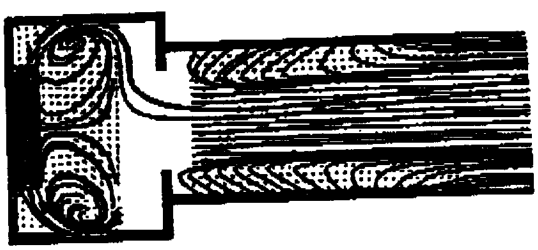


Fig. 11: Configuration 4 LDV streamlines

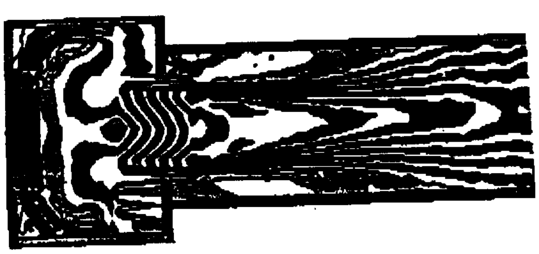


Fig. 12: Configuration 4 LDV velocity contours

Fig. 13. Effect of inhibitor, axial injection

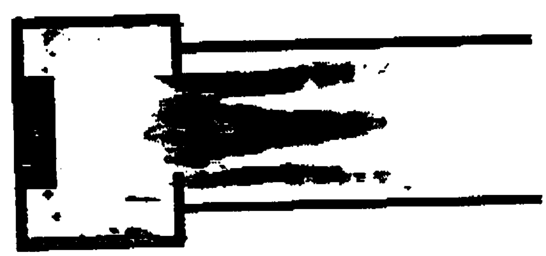


Fig. 14: Effect of inhibitor, radial injection

Summary and Conclusions

An understanding of the basic flow dynamics of the subject hybrid model has been gained through this series of testing. Changing injectors (axial vs. radial) and inhibiting the flow between the upstream plenum and the CP section changes the basic flow structure, as evidenced by streamline and velocity contour plots. Numerous shear layer structures were identified in the test configurations; these structures include both standing and traveling vortices which may affect combustion stability. Standing vortices may play a role in the heat addition process as oxidizer enters the motor, while traveling vortices may be instability mechanisms in themselves. Finally, the flow visualization and LDV measurements give insight into determining the effects of flow induced shear layers. One such case is illustrated in Fig. 7 where the CP wall flow (and possibly the flame front in a hot flow situation) is being drawn into the upstream plenum, whereas Fig. 11 shows that the presence of an inhibitor may reduce this tendency.

When the external oscillation system was used in configurations 1 and 4, no changes in the flow characteristics could be detected visually. The LDV measurements, being time-average values, did not discern any changes either. Therefore, one modification to the model will involve the addition of fifteen (15) fluctuating pressure transducers in various locations on the model. This, coupled with the inclusion of a 45° injector and the 20% and 36% inhibitor plates, will further extend the understanding of the model flow characteristics. Particle displacement tracking (PDT) will also be done with configurations 1-4 and with subsequent test configurations. The results of all testing will be compiled and used to build a CFD model to predict flow behavior. Once the CFD model is in place, hypotheses linking flow dynamics to hybrid motor stability can be tested.

References

[1] Boardman, T.A., Brinton, D.H., and Carpenter, R.L., "An Experimental Investigation of Pressure Oscillations and Their Suppression in Subscale Hybrid Rocket Motors," AIAA Paper 95-2689, July 1995.

[2] Boardman, T.A., Carpenter, R.L., Goldberg, B.E., and Shaeffer, C.W., "Development and Testing of 11-and 24 Inch Hybrid Motors Under the Joint Government/Industry IR&D Program," AIAA Paper 93-2552, June 1993.

[3] "JIRAD 11-Inch Hybrid Motor Test Program," Thiokol Corp. Space Publications, Publication No. 931484, July 1993.

[4] Jenkins, R.M., and Cook, J.R., "A Preliminary Analysis of Low Frequency Pressure Oscillations in Hybrid Rocket Motors," AIAA Paper No. 95-2690, July 1995.

[5] Martin, C., and Van Dyke, M., "Results of an 11-Inch Liquid Oxygen Hybrid Rocket Motor Combustion Instability Study," to be published in JANNAF Conference Proceedings, December 1996.

[6] G.A. Flandro, "Vortex Driving Mechanism in Oscillatory Rocket Flows," AIAA Journal of Propulsion, Vol. 2, No. 3, May 1986, pp. 206-214.

[7] Schadow, K.C., Crump, J.E., Mahan, V.A., Nabity, J.A., Wilson, K.J., and Gutmark, E., "Large-Scale, Coherent Structures as Drivers of Ramjet Combustion Instabilities," 1985 JANNAF Propulsion Meeting, San Diego, CA, April 1985.